Wireless Circuits and Systems Laboratory

Procedure #1 ADS Basics

Contained in this procedure:
1) Creating a project file directory
2) Creating a circuit schematic using lumped components
3) Simulating a circuit to determine its S-parameters
4) Graphing the simulation results

HP ADS (Advanced Design System) Design Type Support

When you first execute HP ADS, or anytime thereafter, you have the option of setting the type of design that is supported. The options are Analog/RF, Digital Signal Processing, or both. The option you should choose is Both, with the Default set to Analog/RF.

1. Creating a New Project and a Circuit Schematic

1. From the Windows START menu, go to Programs/Advanced Design System/Advanced Design System.

2. **If you do not have an existing "project" to work with** (you will only need one project for the entire semester, as each project can contain multiple circuit designs):

   a)   Select File/New Project

   b)   Click on the Browse button to find your home directory where you want the project to be stored (ask your Teaching Assistant for guidance). Type in your project name in the space following the directory reference. You must use alphanumeric characters and underscores (no periods, hyphens, etc.)

   c)   Select the Length Unit to be millimeters.

   d)   Click OK.

Note: A project is a special directory structure within ADS that is used to organize files associated with a particular design task. Each project directory consists of at least five sub-directories – data, mom_dsn (for files associated with the *Momentum* electromagnetic simulation tool), networks, synthesis, and verification - that are created automatically for each new project. The project's structure helps keep related design information grouped together.

At this point the "Schematic" window should open up automatically. If the schematic window does not open automatically, click on the Window -> New Schematic menu or simply the 'New Schematic Window' button near the middle of the main ADS menu. As long as you work in this project directory, all of your circuit designs, circuit layouts, and test benches will be stored in various sub-directories of your project.

3. **If you do have an existing project**:
   a)   File/Open Project

   b)   Change to the appropriate drive location

   c)   Double click on your project directory

   d)   Single click on your project directory again and click OK

At this point the "Schematic" window should open up automatically. If the schematic window does not open automatically, click on the Window -> New Schematic menu or simply the 'New Schematic Window' button near the middle of the main ADS menu.

4. It's good practice to give your schematic design a name right as you begin. To do this, click on File/Save As and then enter the name of your design (you do not have to add the suffix). This file will be saved in the Networks subdirectory of your project directory. For now, let's assume our design is called lpf_1. [If you already have an existing design that you want to open, click File/Open and select the file.]

Note: In the following steps you will be placing several items into the schematic window. Most of these items will have red dots on one or more sides that are 'connection points'. When placing the items, leave ample space between any two adjacent items such that the red dots from one item do not come into contact with those of another. Eventually, we will use the 'wire tool' to manually connect appropriate items together.

5. Now let's layout a low-pass filter that consists of the following: a series inductor, a shunt capacitor and another series inductor. For convenience, you can obtain a menu of common commands by clicking the right mouse button inside the drawing window.

a) On the left side of the schematic window you should observe a collection of lumped elements (resistors, capacitors, inductors, etc.) If this is not the case, click on the pull-down menu near the upper left side of the screen that represents the **Component Palette List**, and select the Lumped Element list. Alternatively, you can click on the **Library** icon (it is a button that resembles 4 books). Select Analog/RF -> Lumped Components on the left side of the library menu---you will then see the list of Lumped Elements on the right-hand side. **For the remainder of this procedure** it will be assumed that you have selected the Lumped Component List from the Component Palette pull-down menu, such that all the necessary elements appear on the left-hand side of the schematic window.

b) Now let's add an inductor to the design. On the left-hand side of the window, the second button down on the far-left represents an inductor. Click on this button, and then drag the mouse onto the schematic area and click to place the inductor. After you place the inductor, you have to 'end' this inductor-placement command. You can either right-click and choose End Command, or click on the large white arrow near the top of the schematic window. (Or, you can simply hit the Escape key.)

c) Now double click on the inductor you just placed into the schematic. Click on the 'L' parameter at the top of the parameter list and set the value to 79.6 nH. The remaining parameters do not have to be modified for this exercise. Click on OK at the bottom of the pop-up menu.

d) Return to the list of icons on the left-hand side of the schematic window and select the capacitor button (the third button down on the left-hand side of the list). Place the capacitor to the right of the first inductor, end the capacitor-placement command, and then double click on the capacitor and change the 'C' parameter to 63.7 pF.

e) In order to make the layout more representative of an actual layout, let's also rotate the capacitor 90 degrees: go to Edit -> Rotate/Mirror and choose Rotate; click on the capacitor and then use the mouse to turn it appropriately. When finished with the rotation, click the large Arrow button on the top row to turn off the rotation mode. After rotating, you can also click on the capacitor and drag it into a better position. Click anywhere on the screen to de-select the object. [An alternative method of rotating an object is to click on the Rotate Items button that is located to the immediate left of the Trash Can (delete) button. After clicking on the Rotate Items button, simply double click on the capacitor.]

f) Finally, add another inductor to the right of the capacitor. Set the 'L' parameter to 79.6 nH. [You can either repeat the previous steps to place the inductor, or click on the Copy Items button that is to the immediate left of the Rotate Items button, and then click on the first inductor and drag the copy into position.]

6. The next step is to add a "Termination" to both sides of the circuit. The type of termination that is used varies with the type of simulation that will eventually be performed. In this exercise we will be calculating the S-parameters of the low-pass filter, and for this we connect 50 Ohm terminations on both sides of the circuit. Click on the Component Palette List pull-down menu and select the **Simulation-S_Param** list. Click on the 'Term' button on the left-hand side of the schematic window (third button down on the right-hand side---it looks like a resistor

with a box around it). Place a Term on the left-hand side of the first inductor and on the right-hand side of the second inductor. (The term number is automatically incremented from 1 to 2 after you place the first termination.)

7. Now we add a "ground" to the lower port of the shunt capacitor and on each termination. The ground button is located near the top of the schematic window, two buttons to the left of the Library. After clicking on the ground button, position the mouse beneath the terminations and capacitor and click once underneath each of them. Remember to turn off the ground-placement function when you are finished (if you didn't do this, you could continue to click and place grounds all over the schematic window).

At this point, your design should look as follows:
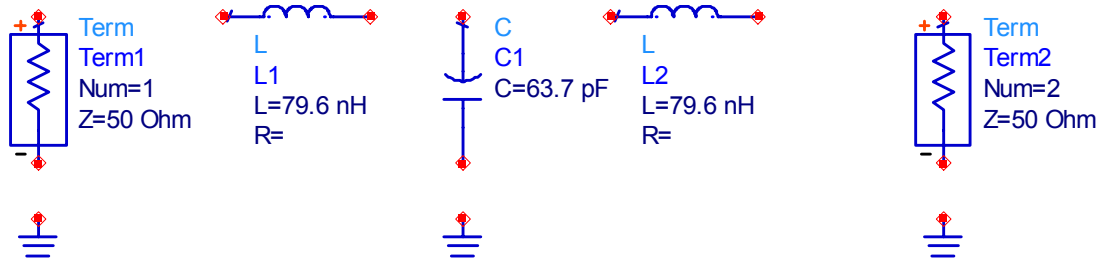
8. To finish the design we need to use the 'wire' tool to connect all the nodes. Click on the Add Wire button near the top of the schematic window (three buttons to the right of the Library). Now simply connect the red dots. For example, click on the red dot on the Ground beneath the first Termination and then click on the red dot at the bottom of the first Termination. Connect the remaining nodes in a similar manner. The completed circuit should appear as shown below:
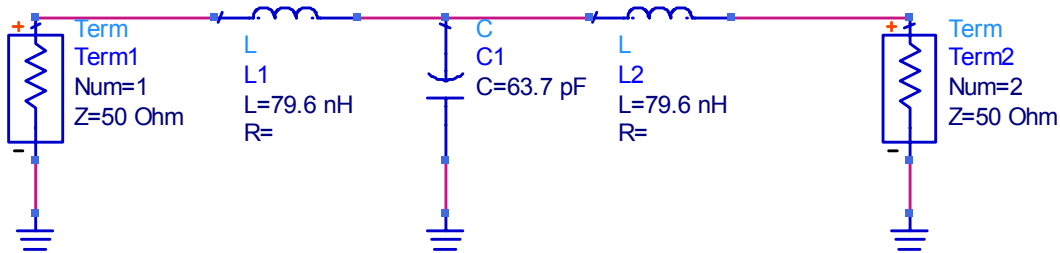
Click on the disk button to save your design.

2. Simulating Your Design

1. Now that we have created the circuit schematic, we can perform a variety of 'measurements' by placing the necessary type of control elements into the schematic window. In this exercise, we will simulate the 2-port scattering parameters.

2. Select the **Simulation-S_Param** list from the Component Palette pull-down menu. Now click on the S_Param: S-parameter Simulation button, that is found on the left-hand side of the schematic window (first button on the far left). Place the mouse within the schematic window and click to insert a copy of the S Parameters block.

3. Double click on the S Parameters block. Change the Start to 0.03 GHz, the Stop to 3.0 GHz and the Step to 0.01 GHz. The number of points will be calculated automatically. (Notice that there are additional tabs for 'Parameters', 'Noise' and 'Display' near the top of the pop-up window. Each tab provides access to options that can be selected to control certain aspects of the simulation, if necessary.)

4. Now, we're ready to analyze the circuit. Click on the Simulate button, which is located five buttons to the right of the Library (books) icon.

When the simulation is initiated, a pop-up window will appear to provide information regarding the status of the simulation. When the simulation is complete, a message will appear indicating that the results have been written to a data file in the Data sub-directory of your current project directory.

3. Creating a Graph of the Simulation Results

1. After analyzing the circuit, click on the New Data Display Window button, which is seven buttons to the right of the Library (books) icon. This will cause a graphics window to pop up.

2. You have several types of plots to choose from. For this exercise, choose the Rectangular Plot, which is represented by the first button below the large white arrow, on the left-hand side of the window. Click on this button and drag the rectangle into the window, and click.

3. To generate plots of S11 and S21 versus frequency, click on these entries under the Data Sets and Equations list, and click the Add button in the middle of the window. Select the dB format for each parameter. Now click on OK to generate the graph.

4. You have several options for changing the default appearance of the plot. If you double click on an individual trace, you can change things like line thickness and color. If you double click on the graph---but not on an individual trace---you can add traces to the graph, change the limits of the x- and y-axes, etc.

5. You can also Print the graph, and/or copy the graph to the Windows clipboard. Your plot (using the default settings) should appear as shown below: